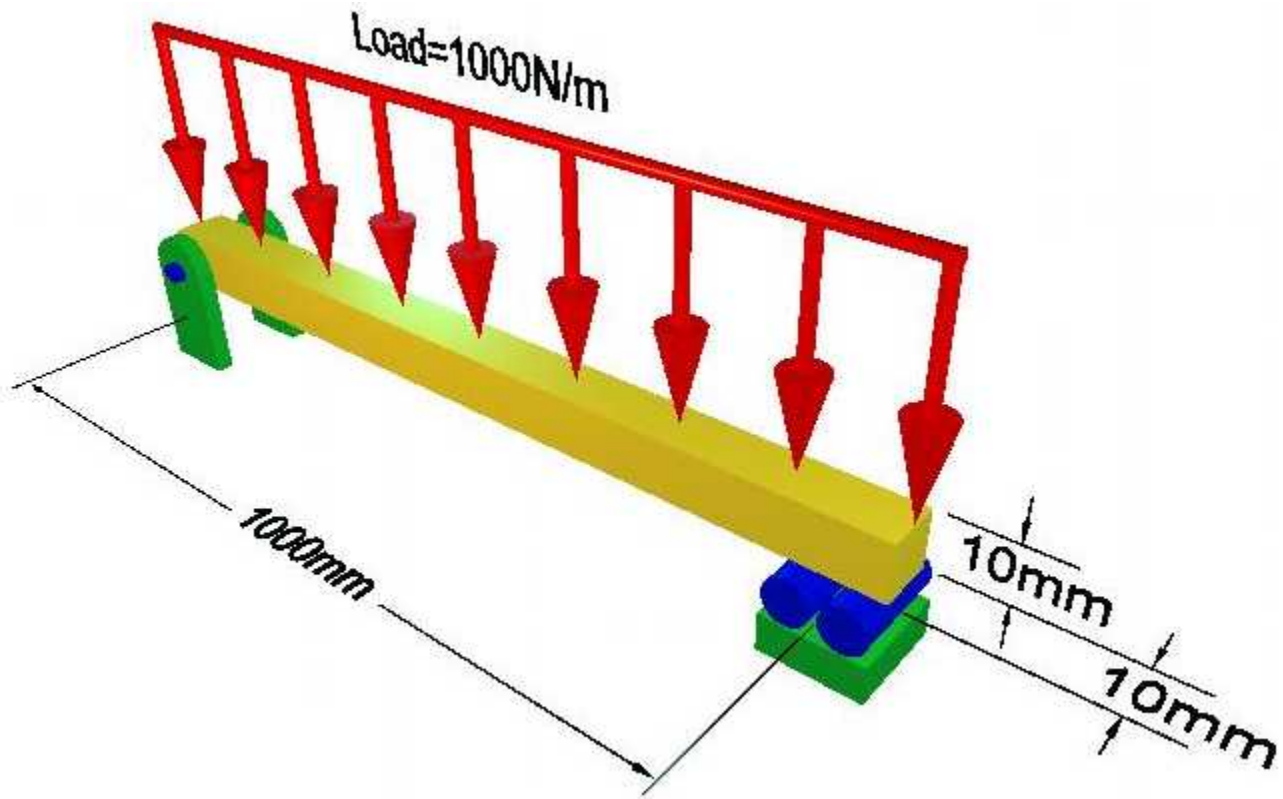


Application of Distributed Loads

Introduction

This tutorial was completed using ANSYS 7.0. The purpose of this tutorial is to explain how to apply distributed loads and use element tables to extract data. Please note that this material was also covered in the 'Bicycle Space Frame' tutorial under 'Basic Tutorials'.

A distributed load of 1000 N/m (1 N/mm) will be applied to a solid steel beam with a rectangular cross section as shown in the figure below. The cross-section of the beam is 10mm x 10mm while the modulus of elasticity of the steel is 200GPa.



ANSYS Command Listing

```
/title, Distributed Loading of a Beam
/PREP7

K,1,0,0           ! Define the keypoints
K,2,1000,0

L,1,2             ! Create the line

ET,1,BEAM3       ! Beam3 element type
```

```
R,1,100,833.333,10          ! Real constants - area,I,height

MP,EX,1,200000             ! Young's Modulus
MP,PRXY,1,0.33            ! Poisson's ratio

ESIZE,100                 ! Mesh size
LMESH,ALL                 ! Mesh line

FINISH
/SOLU

ANTYPE,0                  ! Static analysis

DK,1,UX,0,,,UY           ! Pin keypoint 1
DK,2,UY,0                 ! Roller on keypoint 2

SFBEAM,ALL,1,PRES,1       ! Apply distributed load

SOLVE
FINISH

/POST1

PLDISP,2                  ! Plot deformed shape

ETABLE,SMAXI,NMISC, 1     ! Create data for element table
ETABLE,SMAXJ,NMISC, 3
PLLS,SMAXI,SMAXJ,1,0     ! Plot ETABLE data
```